# Getting Started with CATIA Version 5

# CATIA User Interface

- **Let's review the following key features:**
  - Multi-document support
  - Standard and specific menus & toolbars (File, Edit, Insert, …)
  - Standard manipulation (Copy-Paste, Drag & Drop, …)
  - Intuitive (Highlight, co-pilot, cursor shapes, …)
  - Contextual menu (**M**ouse **B**utton 3)
  - Specification Tree (including all technological features, constraints, relationships, …)
  - Workbenches and associated toolbars …

# CATIA V5 Settings

**For these tutorials you will need to install a material catalogue:**

*Due to the Demonstration restrictions, the data you can use must be flagged. In order to use the material catalogue, you have to replace the code file by the flagged one. Please do the following:*

- *Copy the ..\Getting Started\Catalog.CATMaterial file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials\French directory*
- *Copy the ..\Getting Started\Catalog.CATMaterial file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials\German directory*
- *Copy the ..\Getting Started\Catalog.CATMaterial file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials\Japanese directory*
- *Copy the ..\Getting Started\Catalog.CATMaterial file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials directory*
- *Answer Yes in order to replace the old catalogue*

# CATIA User Interface

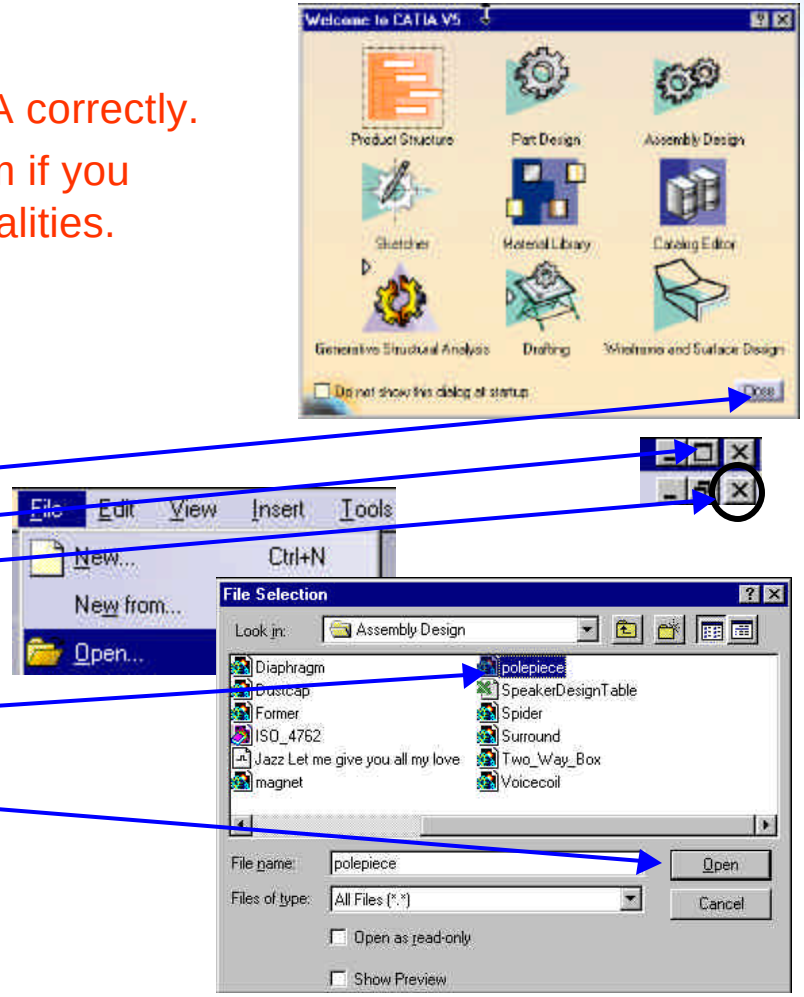
- **IMPORTANT**

✍The next pages are very important to use CATIA correctly.

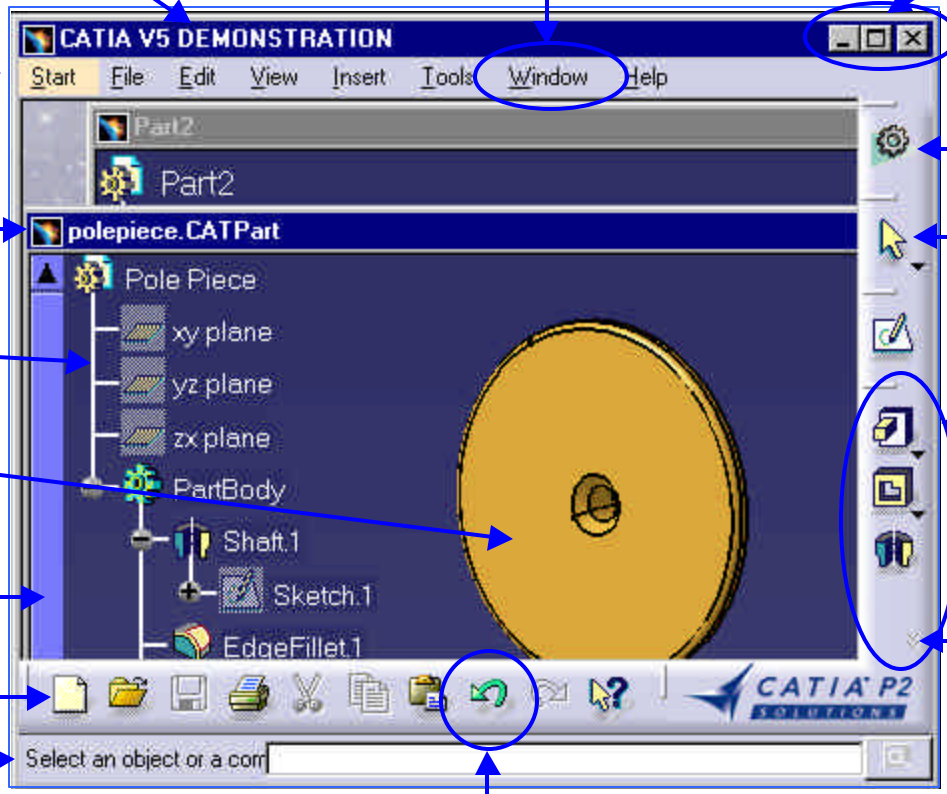
✍You MUST READ these pages and refer to them if you don't remember how to use some of the functionalities.

✍First, open a CATIA file.

- ✍*Launch CATIA if it is not already launched*
- ✍*Click on the Close button*
- ✍*Maximise the CATIA window*
- ✍*Close the Product1 window*
- ✍*Open the file polepiece.CATPart under the …\DATA\Assembly Design\ directory*
- ✍*Select the polepiece.CATPart file*
- ✍*Click on the Open button*



# CATIA User Interface




CATIA window

Use the Window menu to list & swap open windows

Icons to minimise, maximise or close window

Menus

Window of active document

Tree

&

Geometry

Tree scrollbar

Toolbars

Prompt zone

Icon of active workbench

Use this icon to quit a function

Toolbars of active workbench

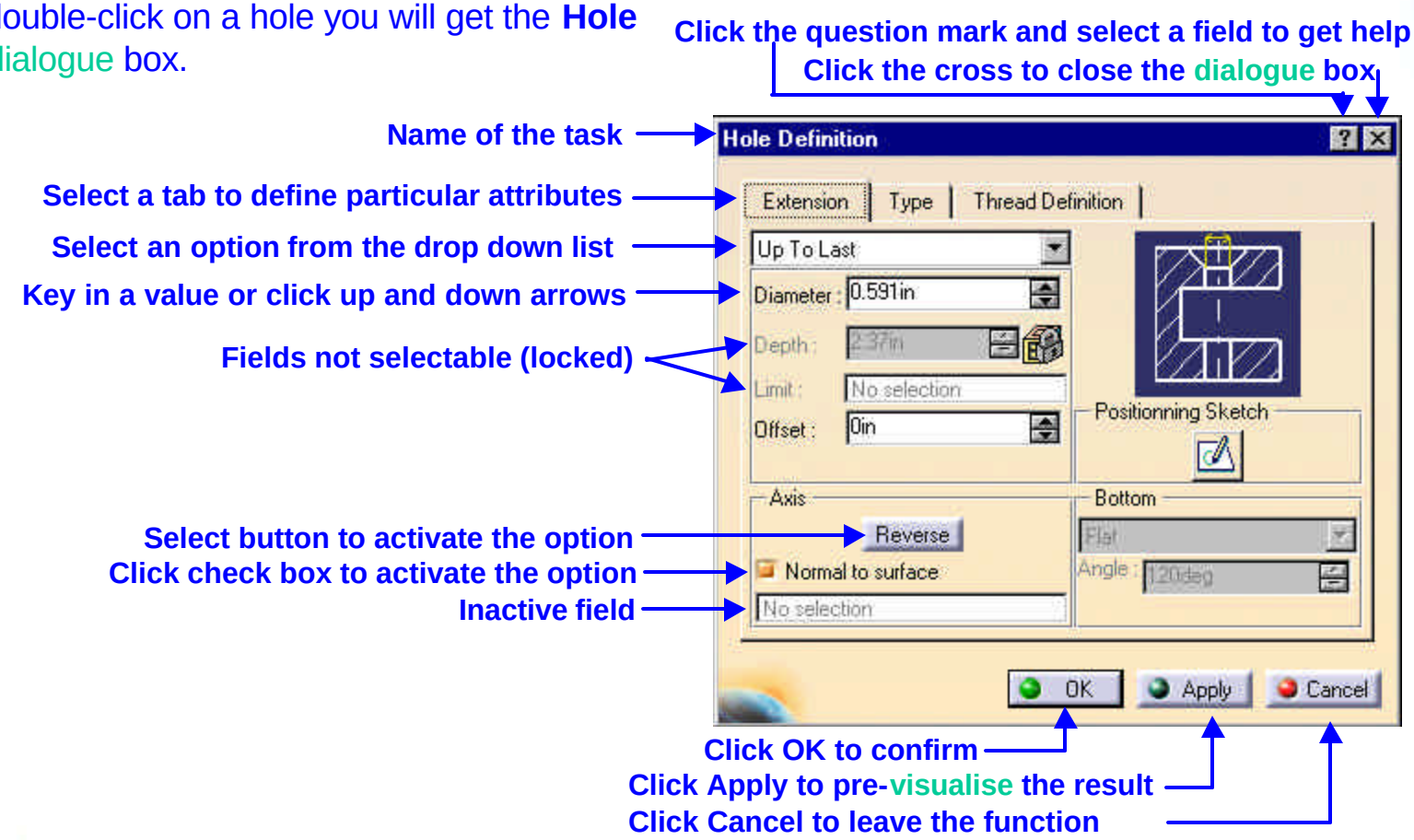
Double arrow indicates more toolbars

Drag the >> sign to move unseen toolbars

Undo icon to cancel actions

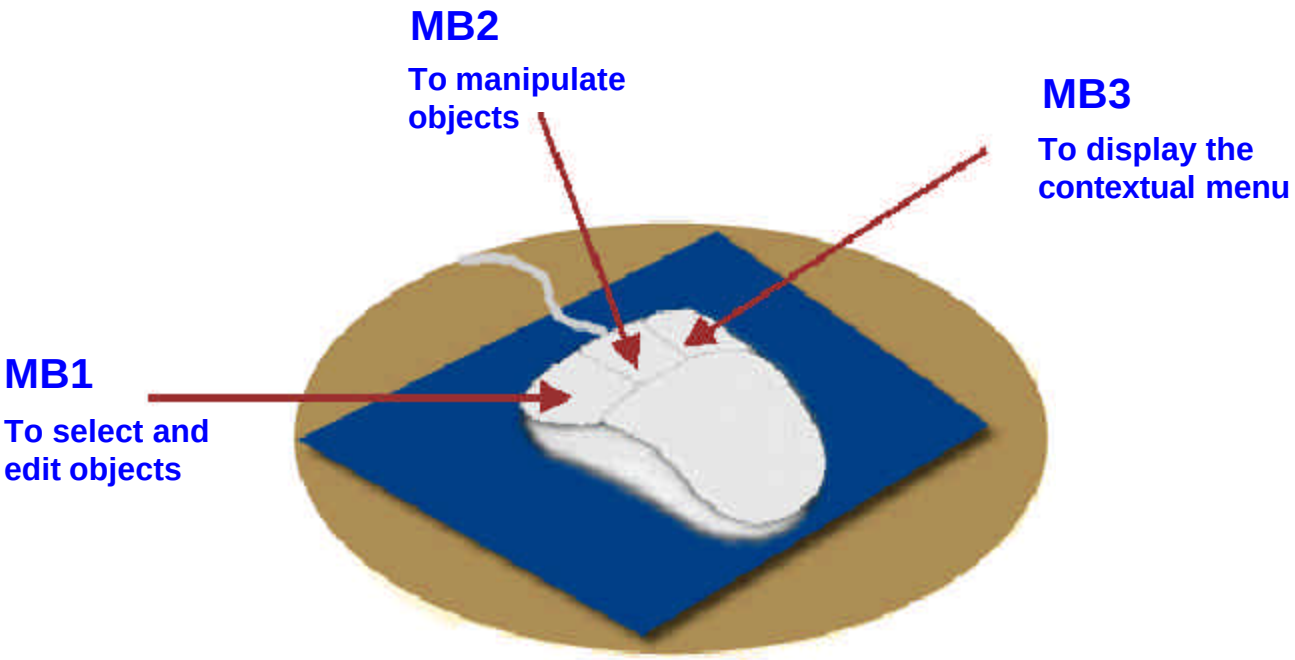
# CATIA User Interface

- **Dialogue boxes provide parameters for the definition of features. They are standardised across all functions.**
  - ✍EX: if you double-click on a hole you will get the **Hole Definition** dialogue box.



# CATIA User Interface

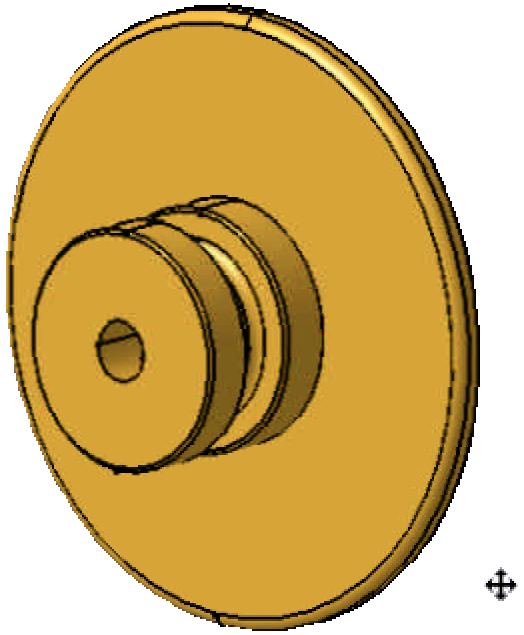
- Using the Mouse Function




***MB*** **means Mouse Button**

- **Panning Objects**

## It's important you practice using the mouse.



1. *Position the cursor anywhere on the screen*
2. *Press and Hold Mouse Button 2 (MB2)*
3. *Move the mouse where you want to drag the part*
   - The cursor will change to a cross

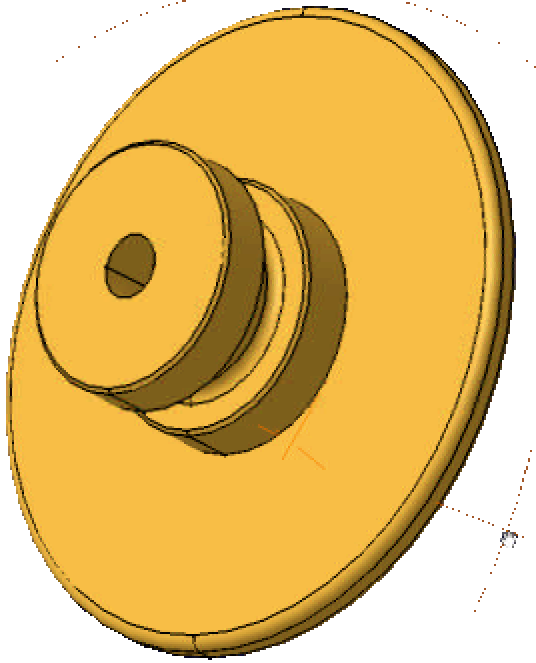
If the part disappears, get it back with the Fit-All-In icon in the View toolbar.

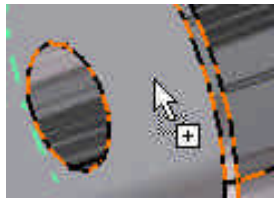
# CATIA User Interface

- *Rotating Objects*

  - *1. Position the cursor anywhere on the screen*
  - *2. Press and Hold mouse button 2 (MB2)*
    - The axis appears in the centre of the screen
  - *3. Press and Hold mouse button 1 (MB1) while keeping MB2 pressed*
    - The cursor changes to a hand and a red circle appears representing a virtual space ball
  - *4. Move the mouse to rotate the part*
    - The cursor will change to a cross

    - *To change the rotation axis, click once on MB2 on the element that you want at the centre of the screen.*
    - *P.S.: if you see a « + » or a Forbidden sign it means that you have not held MB1 and you will duplicate the geometry*

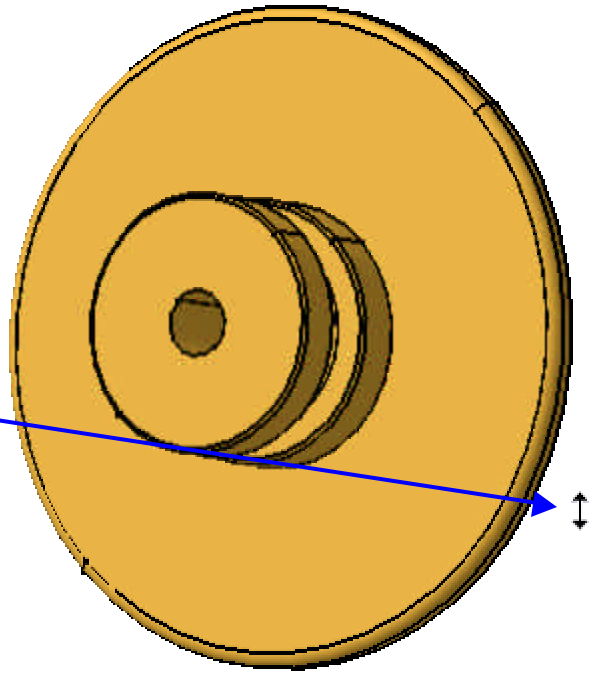




# CATIA User Interface

## •Zooming

- *1. Position the cursor anywhere on the screen*
- *2. Press and Hold mouse button 2 (MB2)*
  - An axis and a circle appear in the centre of the screen
- *3. Press and release MB1 while keeping MB2 pressed*
  - The cursor changes to a double arrow
- *4. Move the mouse up to zoom in*
- *5. Move the mouse down to zoom out*



- To zoom step by step click on + and – icons in the View toolbar

## •Moving the Tree

### ✍Using the Scrollbar

- ✍ ***When <u>and only when</u> the size of the tree exceeds the window you can use the scrollbar to display the tree downward or upward***

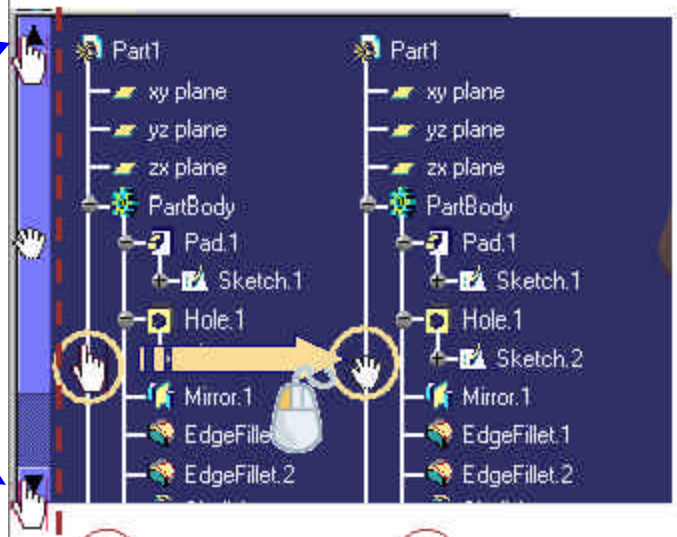
**Click up arrow to display the top of the tree**

**Drag the bar up and down as you wish**

**Click down arrow to display the bottom of the tree**

### ✍Using the Mouse

- ✍ *Moving the tree anywhere*




1. **Position the cursor on a tree branch.**

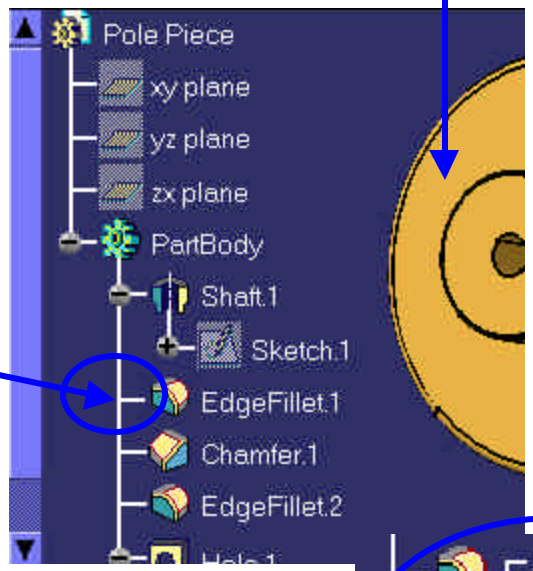
   **The cursor changes to a pointing finger**

2. **Click mouse button 1 (MB1) and drag the mouse to move the tree.**

   **The cursor changes to a small hand**

## •Adjusting and Expanding the Tree

**Geometry dimmed while adjusting the size of the tree**




1

**Click mouse button 1 (MB1) on a tree branch**

2

**Zoom in and out, as explained two pages before**


3

**To reactivate the geometry, click again on a tree branch**

**To adjust tree size:**



**Click on + to expand**
**Click on – to shrink**

**Expand / Shrink**

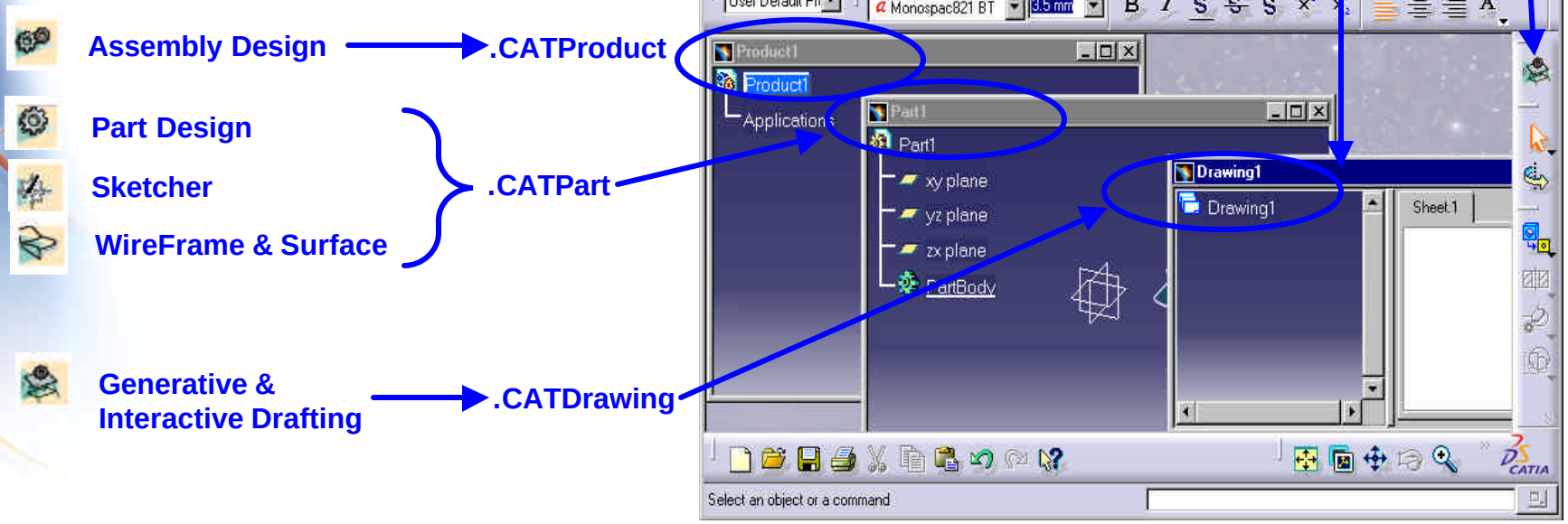
*Note that you can use the scrollbar to display the top or bottom of the tree*

# CATIA User Interface

## CATIA Documents

- When working in CATIA you can create, modify and save geometries within documents
- Documents are containers
- **Different Types of Documents**

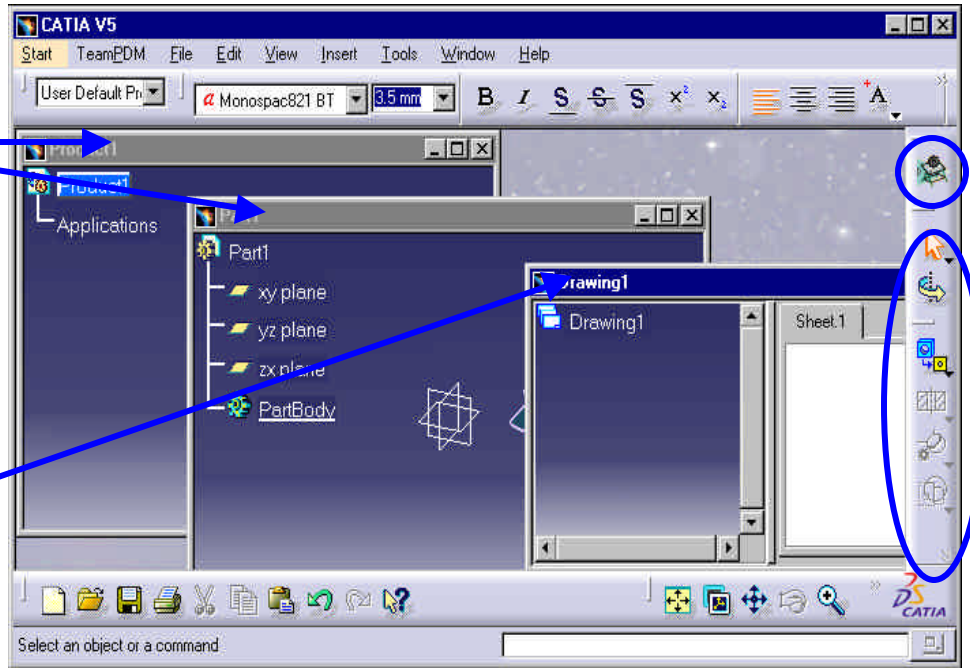
The type of document depends on the workbench you use to create and modify a geometry :



# CATIA User Interface

## •Terminology

- A Product Structure is a way to structure and organise your products logically. You can navigate within the structure and work on its different components
- A Document is a file including data you can create and manage with the associated workbench (specific extension)
- A Workbench is a set of tools that allows you to create and manage your geometry



# SKETCHER Workbench Presentation

# Learning Sketcher

- **Exercise**

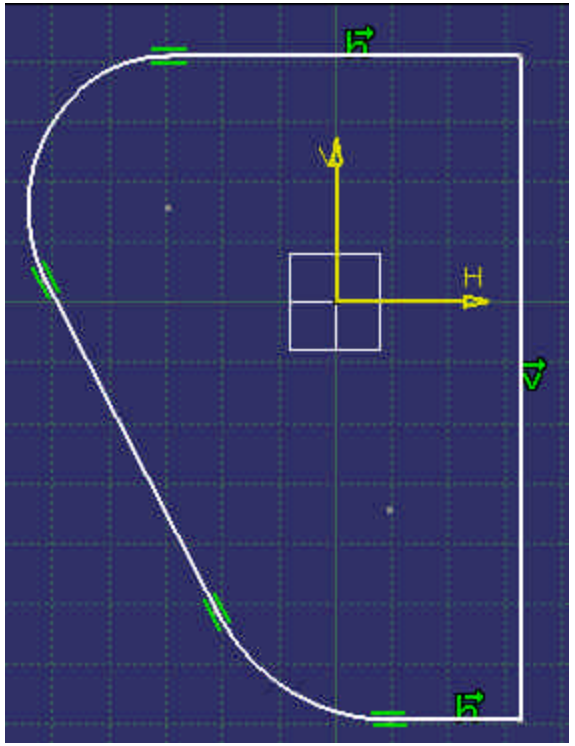
✍ Open a new Part : Start + Mechanical Design + Part Design

**1** **Select a plane, a Solid face, or a Planar Surface to Sketch on**


**2** **Access the Sketcher Workbench**

**3** **Select the Polygon icon and draw a sketch as shown**

- ✍ Click and release MB1 where you want to start a line
- ✍ Click MB1 where you want to finish the line
- ✍ Hold MB1 then move the mouse and release MB1 to draw an arc
- ✍ Double-click when you want to finish your profile




✍ *If you want to exit the function click on the* ***Select*** *icon*

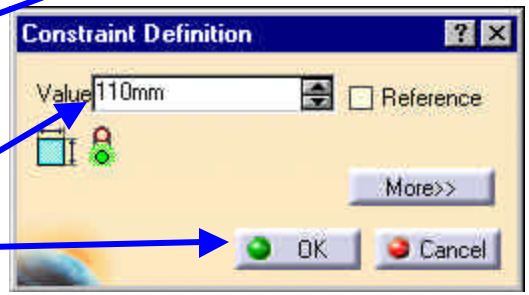
- **Exercise**

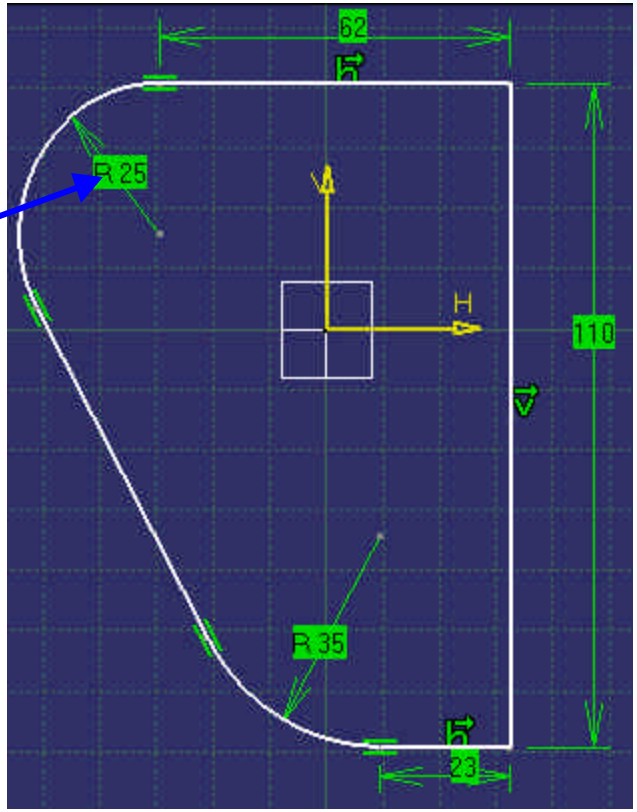
**4** **Select the Constraints icon and set dimensions**

- To set a dimension, select a line, drag the dimension to the desired position
- Click to create it

**5** **Change Dimensions as shown**

- To modify a Dimension, double-click on the green dimension value box.
- Change the value and click OK to validate




**6** **Exit the Sketcher**

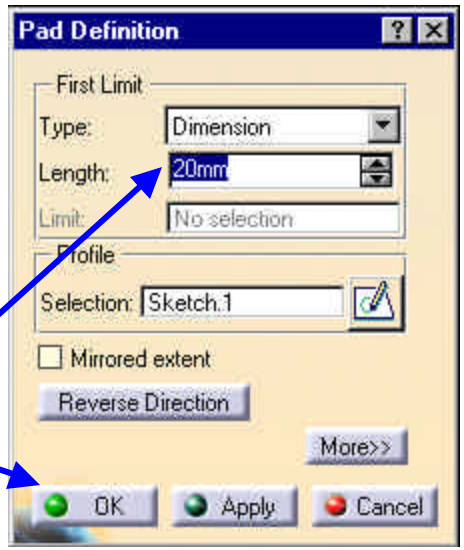
- **Exercise**

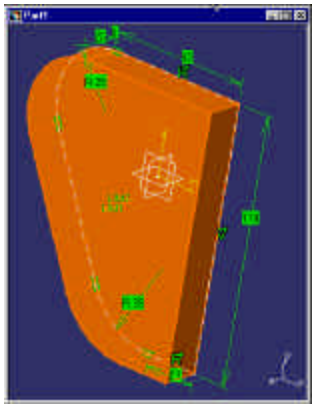
**7** **Click on the Pad icon** 

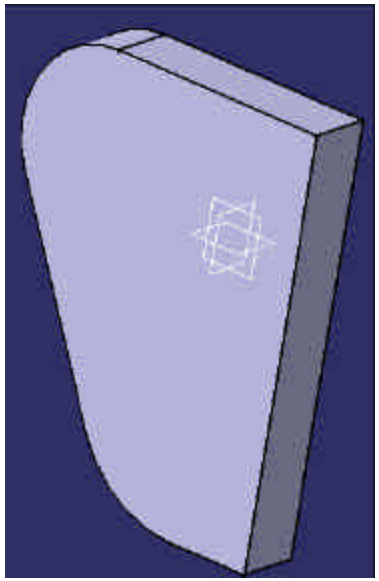
✍ *Select the Sketch if necessary*

**8** **Click OK to validate**

✍ *You can change the length value before validation*



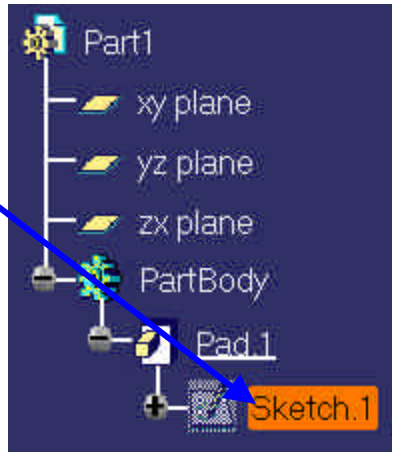



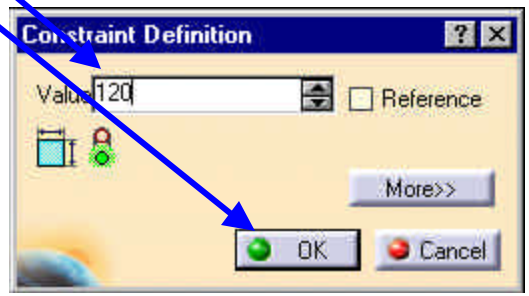


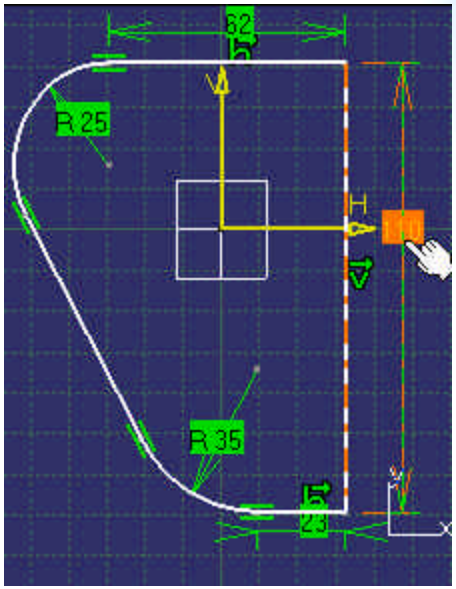
- **Exercise**

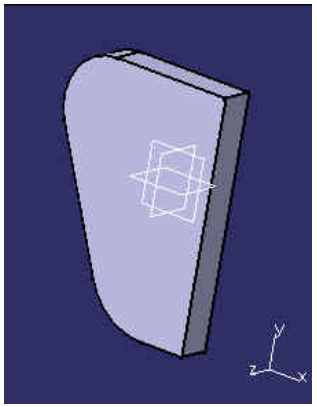
**9** **Double-click on Sketch.1 in the Tree to enter Sketcher**

- Double-click on the 110 dimensions
- Change the value to 120
- Click OK to validate
- Exit the Sketcher 



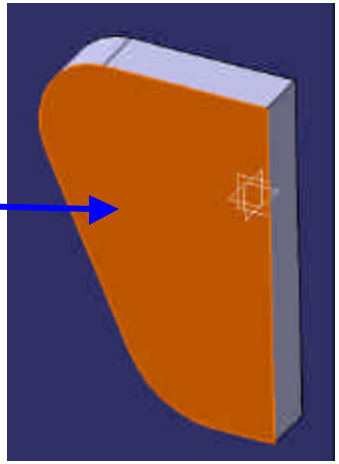



# Learning Sketcher

- **Exercise**

**10** **Select the face as shown and enter Sketcher**



**11** **Select the Circle icon**

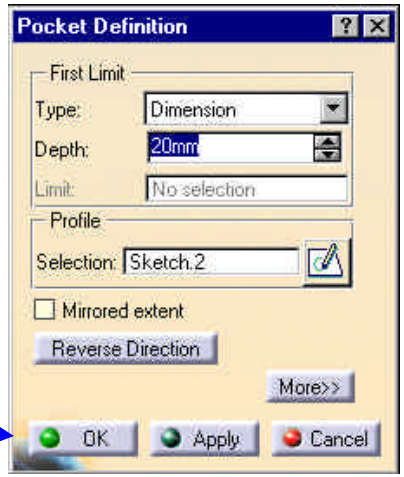
- Click to define the circle centre
- Drag the mouse to define the radius
- Click to create the circle

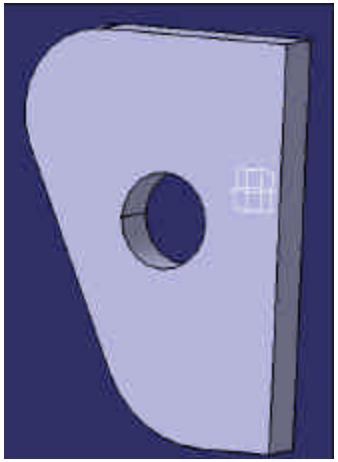
**12** **Exit the Sketcher**

**13** **Select the Pocket icon**

**14** **Click OK to validate**

- *You can change the depth value before validation*



**15** **Close all the windows except the CATIA one. This is the END of the Sketcher Training**

# SKETCHER Workbench Presentation

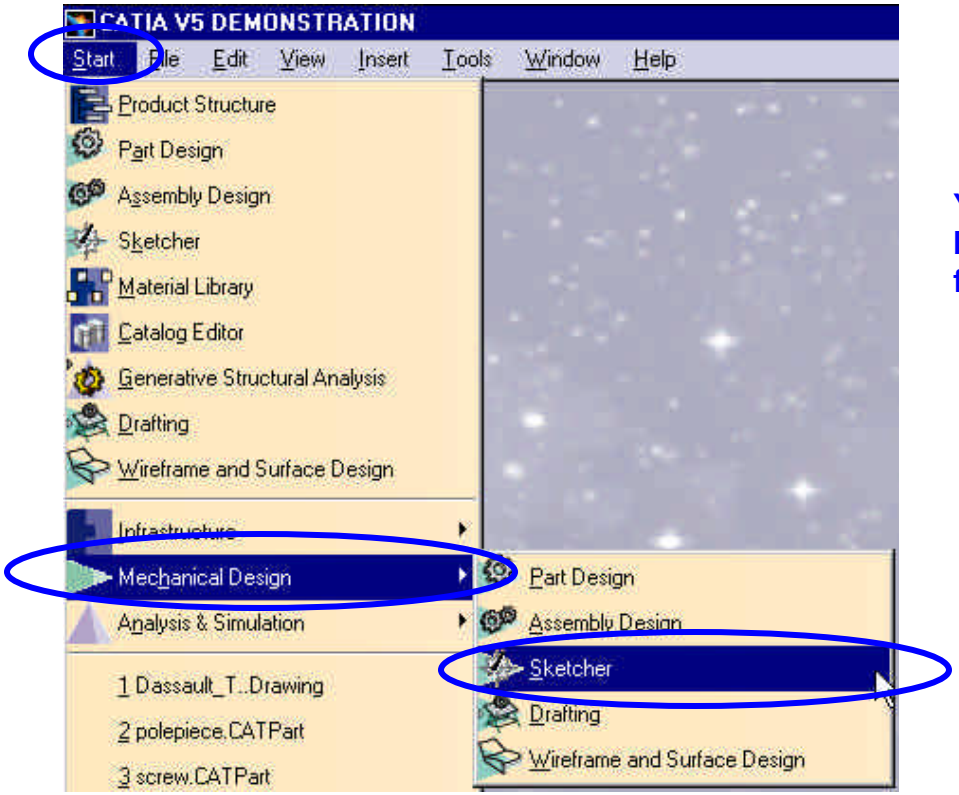
- **Let's have a short Debriefing…**

  - The Sketcher is a set of tools to help the user quickly generate 2D geometry
  - The completed Sketch can then be used to generate solids and surfaces
  - The capability to define constraints between elements in the Sketcher allows for quick modifications of the Sketch on subsequent solids or surfaces
  - Others tools such as Animate Constraints enable the user to explore design alternatives

# Learning Sketcher

- **Accessing the Workbench**

  - Select Start + Mechanical Design + Sketcher
  - Select a plane or a face of an object

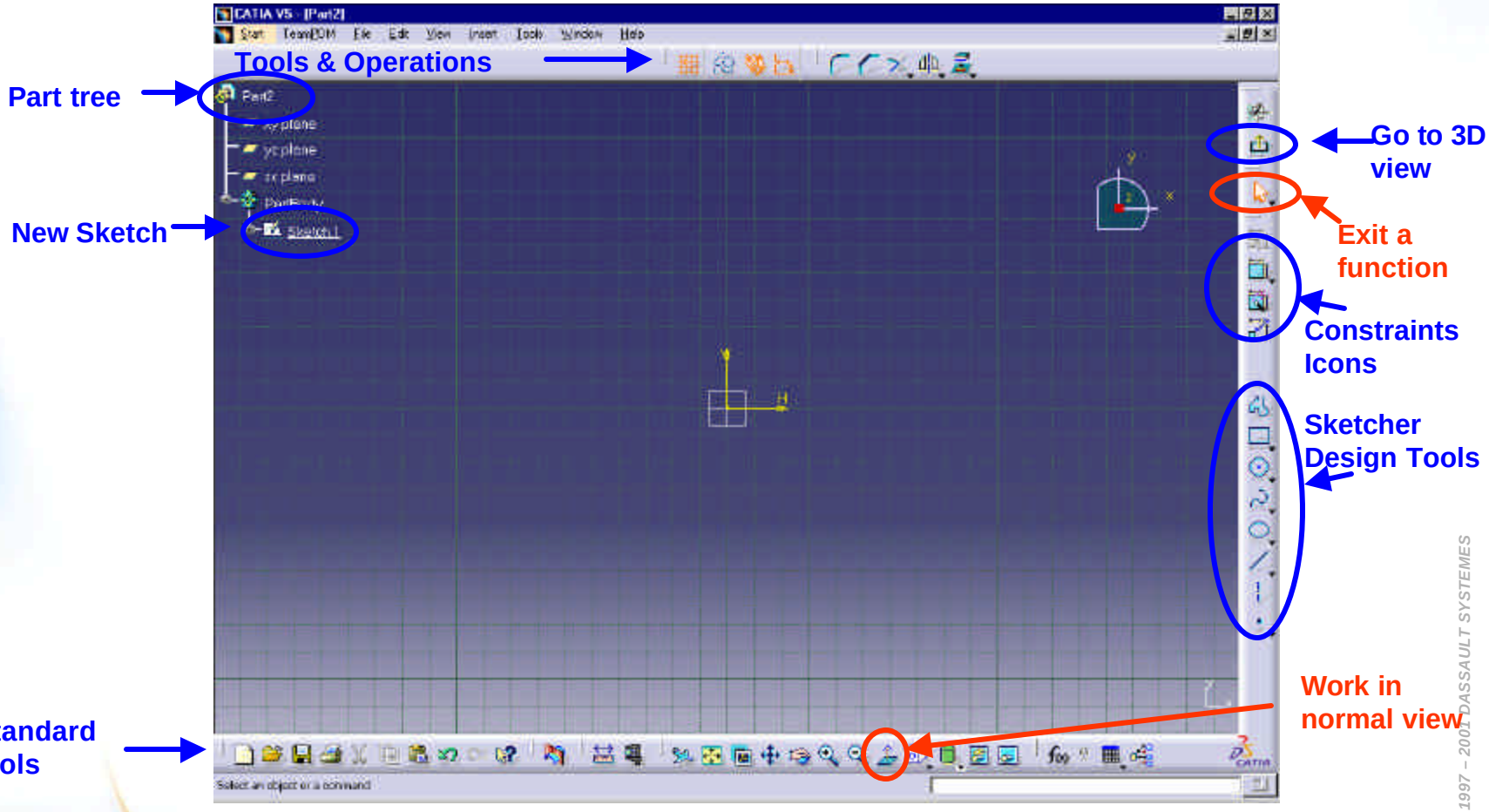



**You can also access the Sketcher by selecting the Sketcher icon from any Workbench**

# Learning Sketcher

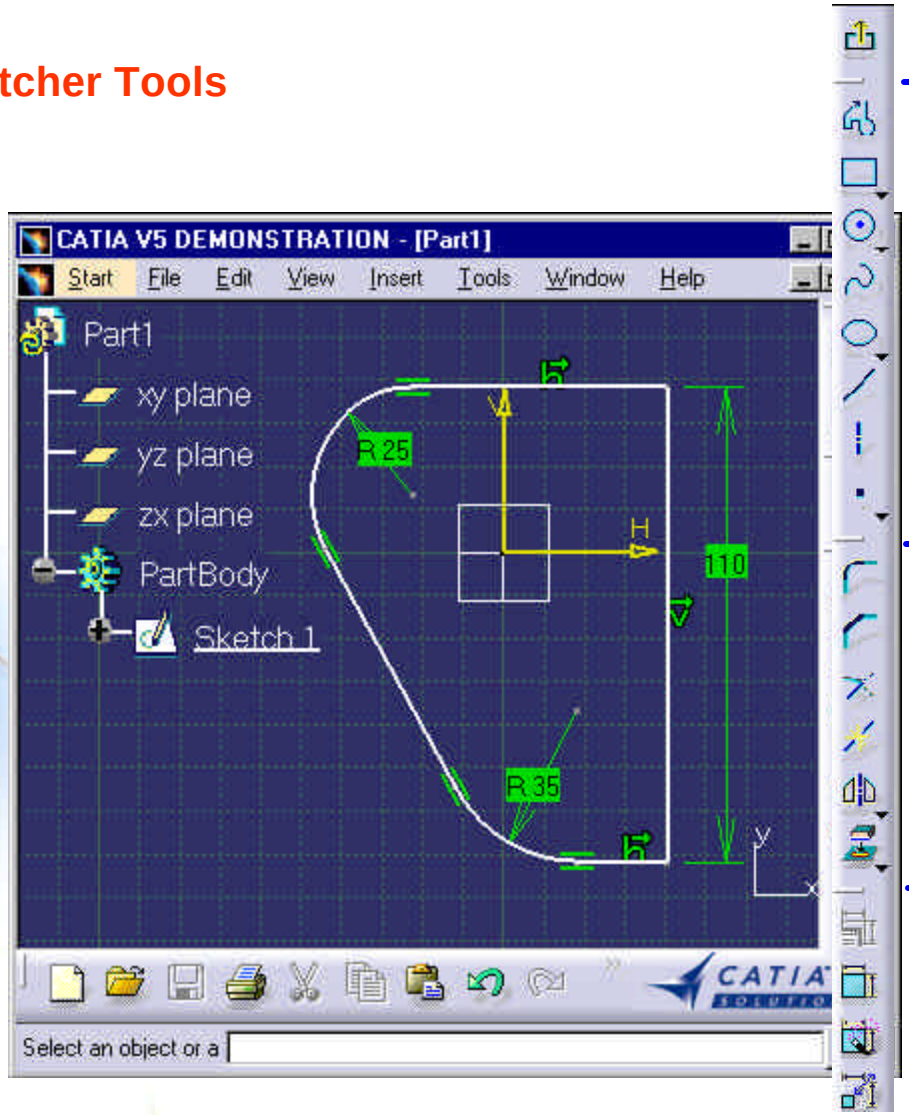
- **Sketcher Workbench overview**

✍A new Sketch will appear in the Part Tree when entering the Sketcher Workbench



# Learning Sketcher

- **Sketcher Tools**




**Exit Sketcher**

**Profile**

**Rectangles, Keyholes, Polygons…**

**Circles, Ellipses, Arcs…**

**Spline**

**Ellipse**

**Line**

**Axis**

**Points…**

**Corner**

**Chamfer**

**Trim options…**

**Break**

**Symmetry**

**Projection**

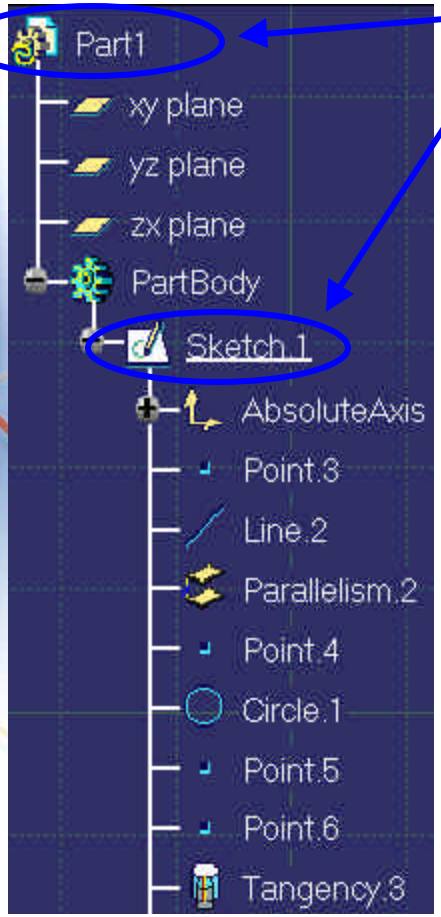
**Constraints dialogue box**

**Constraints**

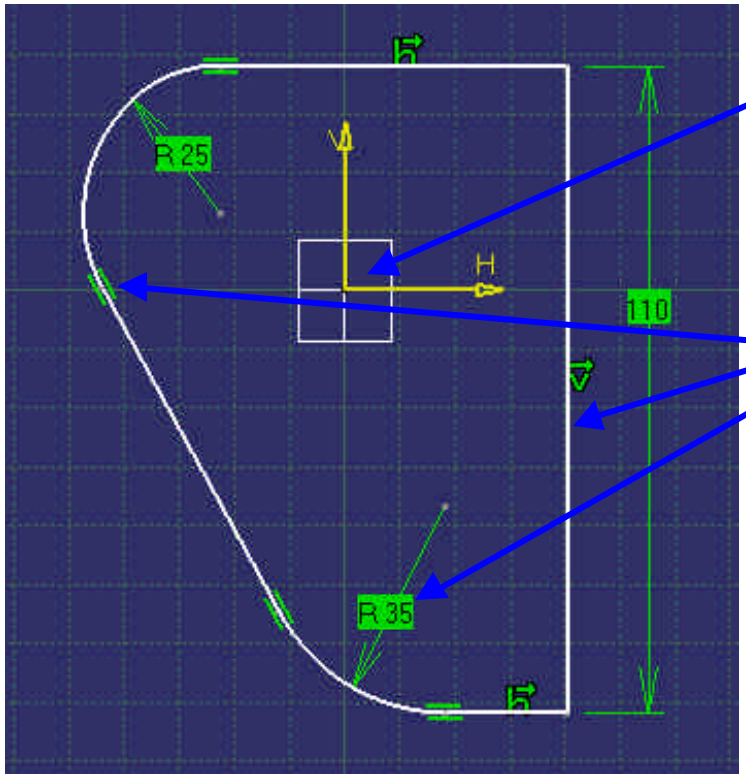
**Auto Constraints**

**Animate Constraints**

- **Terminology**




**The Sketch is the holding place for a group of 2D elements on a specific plane. There can be more than one Sketch using the same plane as support.**




**The V-H Axis is the origin of the Sketch.**

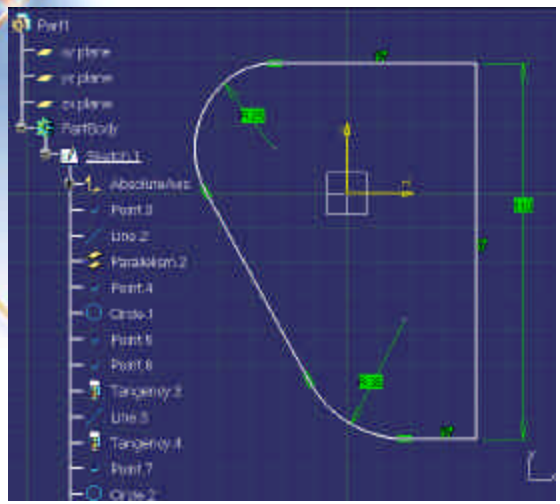
**Sketches generally consist of a Profile, Constraints, and Dimensions (a type of Constraint).**

- **General Process**

1. Select a plane, a Solid face, or a Planar Surface to Sketch on

2. Access the Sketcher Workbench 


3. An In-Work Sketch is added to the Specifications Tree

4. Create Geometric Elements and Constraints

5. Exit the Sketcher 

   Use the Sketch to create a Solid or a Surface